Dimensioning

I-DEAS® Tutorials: Fundamental Skills

Learn how to:

- create dimensions
- modify dimension values
- add a 2D fillet
- find free degrees of freedom

Before you begin...

Prerequisite tutorials:

1. Getting Started (I-DEAS™ Multimedia Training)

–or–

Introducing the I-DEAS Interface,
Quick Tips to Using I-DEAS
–and–
Creating Parts

2. Sketching and Constraining

To begin this tutorial, make sure you're in the following application and task:
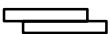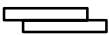
Design, Manufacturing, or Simulation

Modeler

If you saved the model file created in the tutorial "Sketching and Constraining," open it now and use the sketch that was created.

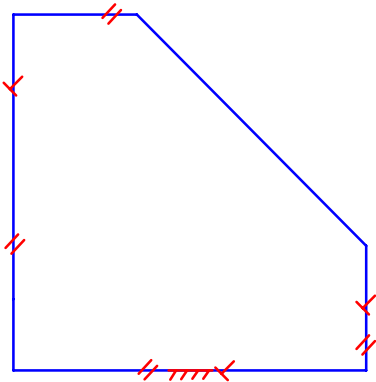File
    Open

Set your units to mm.

Options
    Units

mm (*milli newton*)

If you did not save the model file, or the sketch is no longer on the workbench, create the following sketch before continuing. Delete any dimensions, if necessary. You'll learn how to recreate them in the next steps.

Save your model file.

File
   Save

Warning!

If you are prompted by I-DEAS to save your model file, respond:

No

Save only when the tutorial instructions tell you to—not when I-DEAS prompts for a save.

If you make a mistake at any time between saves and can't recover, you can reopen your model file to the last save and start over from that point.

Hint

To reopen your model file to the previous save, press Control-z.

Dimensions are a special form of constraints. They allow you to control and modify distances between points and lines, while maintaining other geometric constraints and other dimensions.
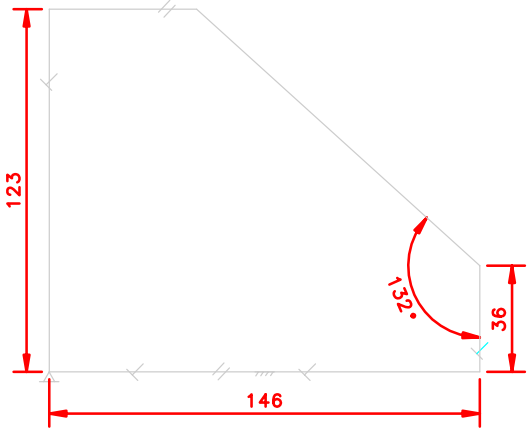
There are many ways to create dimensions:
- point-to-point
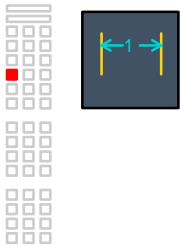- point-to-line
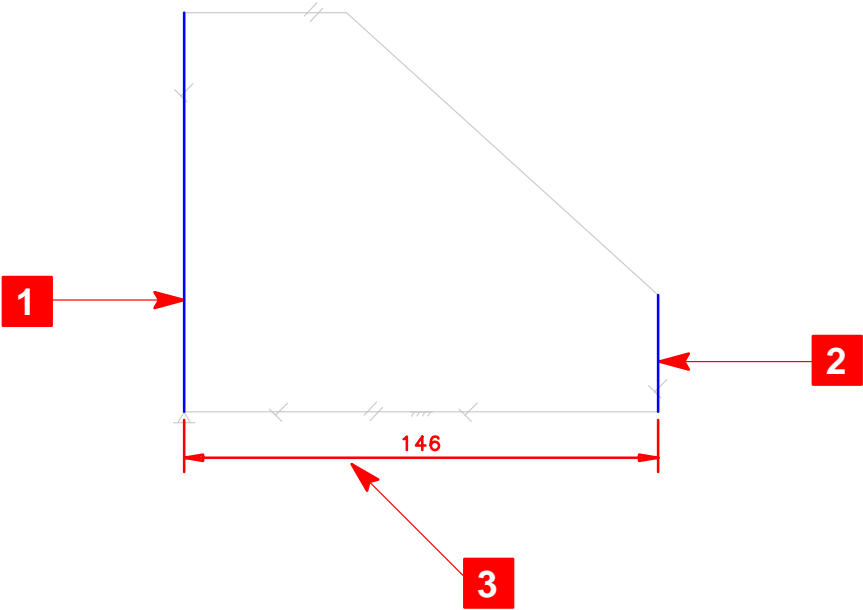- line-to-line

With options:
- *Horizontal*
- *Vertical*

These options are discussed in this section.

Create a line-to-line dimension like the one shown. Don't worry about the actual value. You will modify dimensions later in the tutorial.
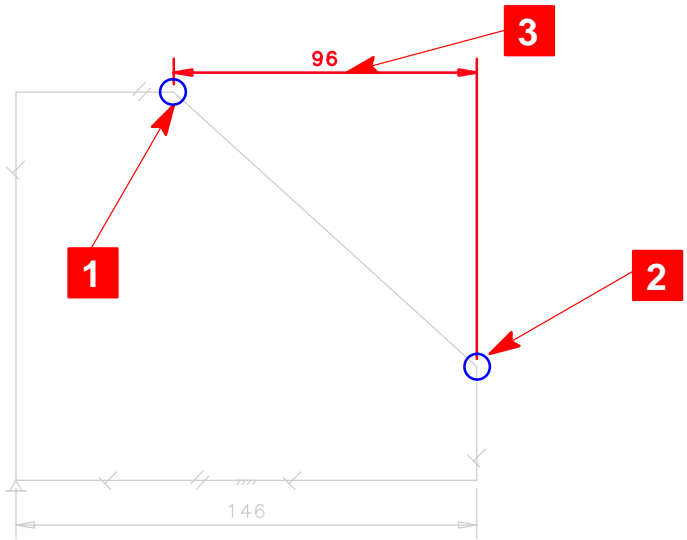
1 pick line

2 pick line

3 place the text

Create a point-to-point dimension, using the *Horizontal* option.

1 pick point

2 pick point

Horizontal

3 place the text

Things to notice

As you add dimensions, the wireframe will change colors:

- green = unconstrained
- yellow = partially constrained
- blue = fully constrained

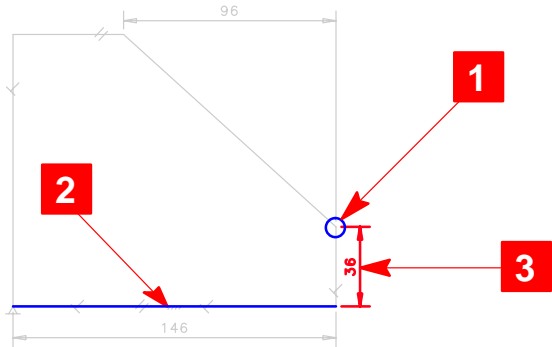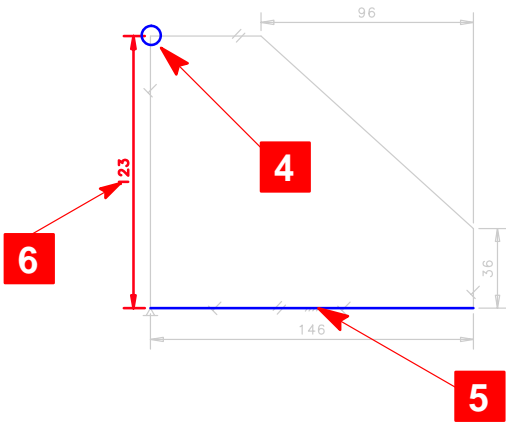The *Horizontal* and *Vertical* options allow you to control point-to-point dimensions by selecting the horizontal or vertical component of the angled distance.

Create dimensions

Create the two vertical dimensions as shown. Create these by picking a line and a point both times.
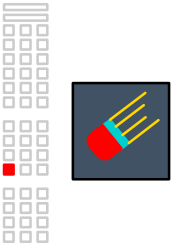
1 pick point

2 pick line

3 place the text

4 pick point

5 pick line

6 place the text

Delete the dimension on the angled face so you can add an angular dimension instead.
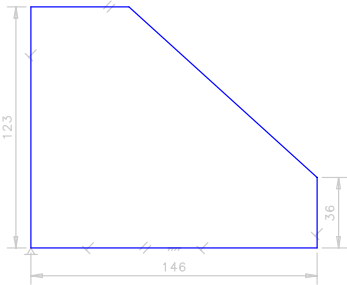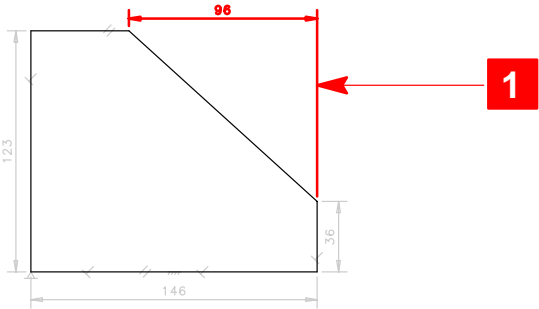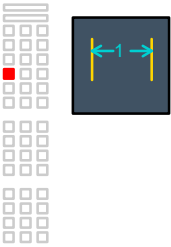

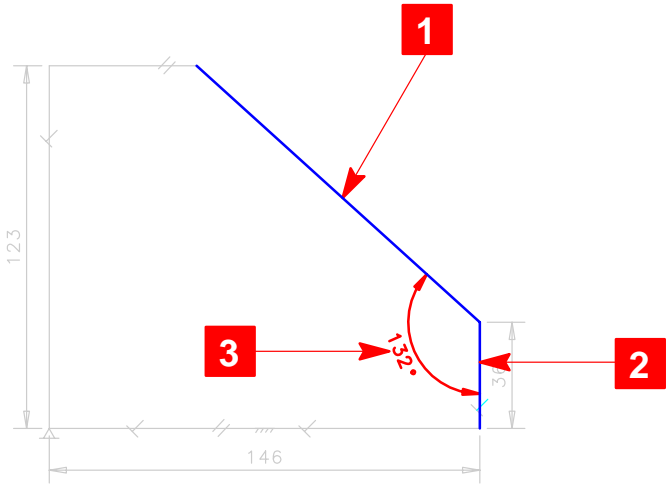**1** pick dimension

 (Done)

 (Yes)

 (to deactivate icon)

Now, add the angle dimension.

1 pick line

2 pick line

3 place the text
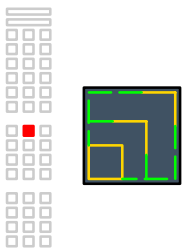
Recovery Point
File
Save

One of the benefits of I-DEAS is that you can sketch the shape and then later modify the dimensions to the size required.

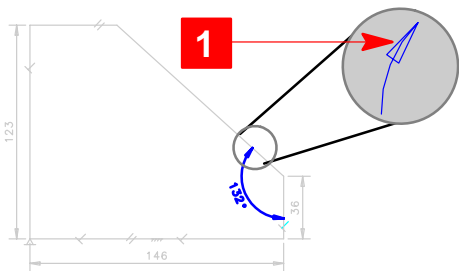In this section, you learn different ways to modify dimension values.

You can modify dimensions with the:

- *Drag* command
- *Modify* command

Use the *Drag* icon to change the angle to135 degrees.
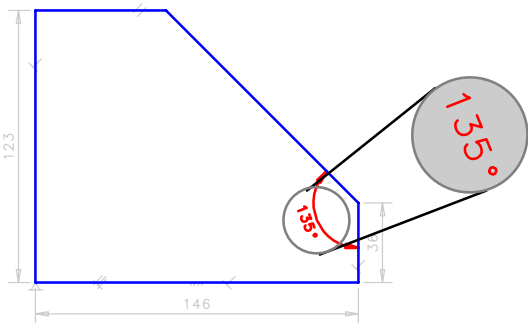

1 pick arrowhead

Hint

After picking the arrowhead, move the angled line while watching the degrees in the odometer in the graphics window. When it reaches 135, click the left mouse button.
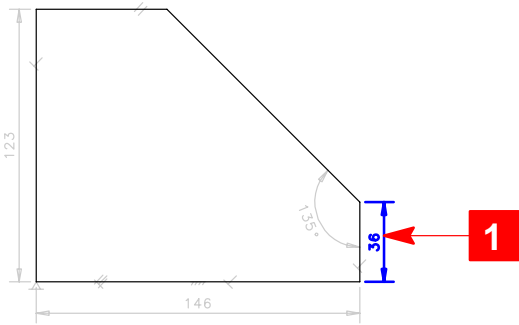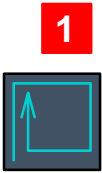
For dimensions, you can drag:

- an arrow to modify the size of the dimension
- the text to change the location
- a leader to reconnect the dimension to a new location
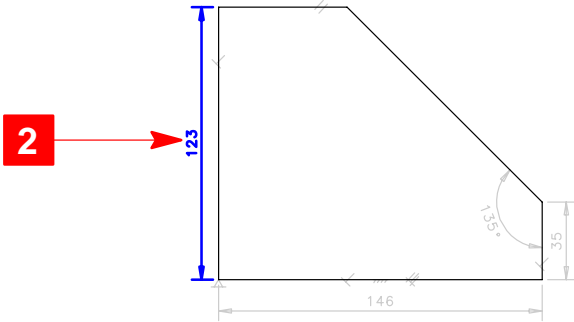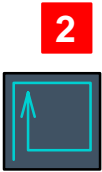

 (to deactivate icon)

Use the *Modify* icon to directly modify the two vertical dimensions as shown.

1

Modify Dimension form

= 35

OK

2

Modify Dimension form

= 100

OK

Use the *Modify* icon to match the horizontal dimension to the longer vertical dimension.

◻◻▮ *Deselect All*

1

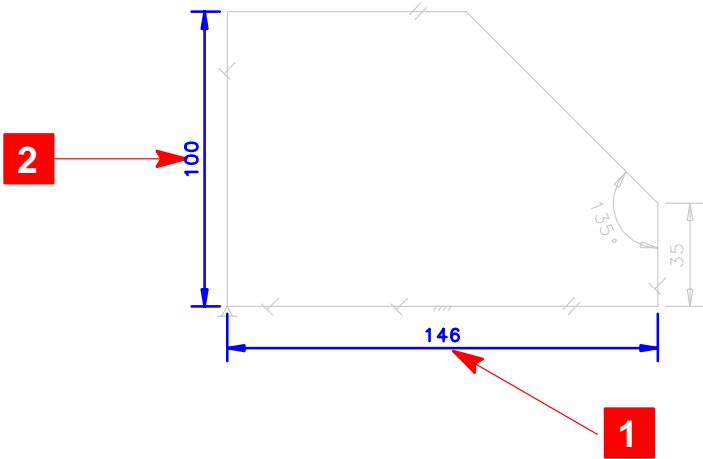## Modify Dimension form

→ *Match*

2

▢ *OK*

◻▮◻ (Done)

Things to notice
The horizontal dimension that is now matched to the vertical dimension is identified with brackets drawn around it.
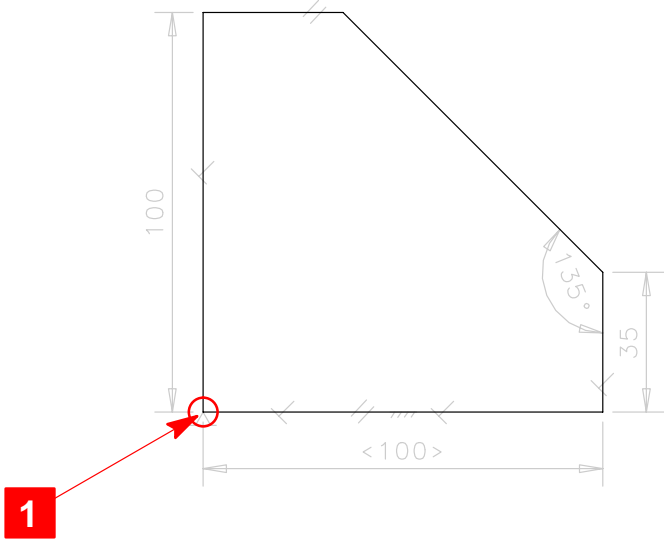
When you create a fillet, it is automatically constrained with a tangent constraint on both ends and a radius dimension. However, you can control the radius and optionally leave the untrimmed curves in place. These untrimmed curves are useful for dimensioning to the "theoretical corner."
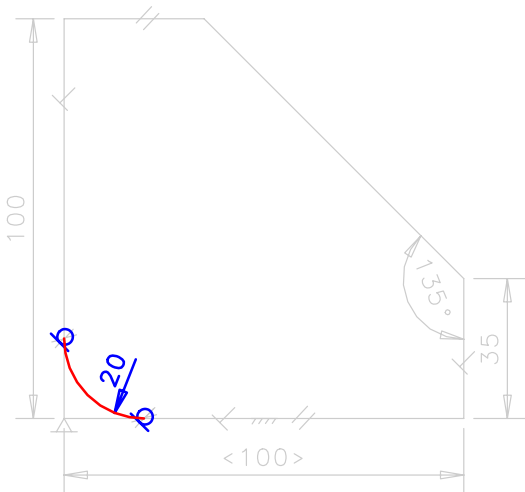
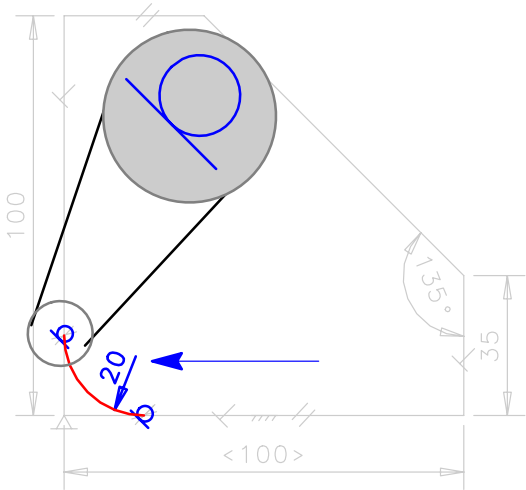Add a fillet in the lower corner. Keep the untrimmed curves.
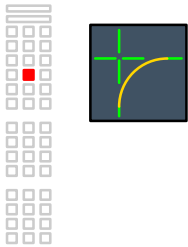
1 pick corner

Fillet form

Radius: 20

*Trim/Extend* (toggle off)

OK

Things to notice
Constraints (tangent) and dimensions (radius) are automatically created.

In any 2D drawing, there must be a certain number of dimensions to fully define a part so it can be manufactured. More dimensions would be redundant, while fewer dimensions would leave ambiguities.

The same holds true with any 2D sketch. If the shape is not fully constrained, unpredictable results may occur when you make modifications.

This section shows you how to determine if you have any unconstrained geometry.
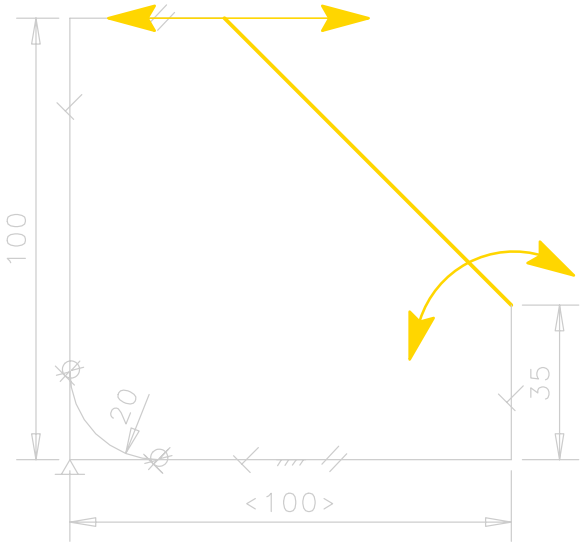
To demonstrate how to find free degrees of freedom, delete the angular dimension so that the geometry is under-constrained.
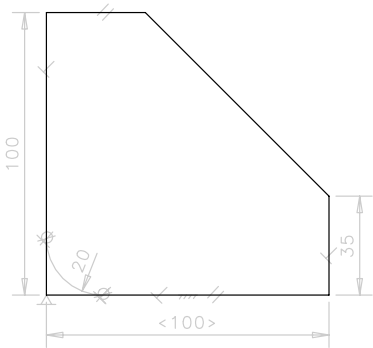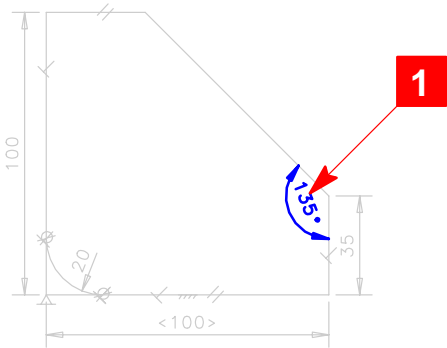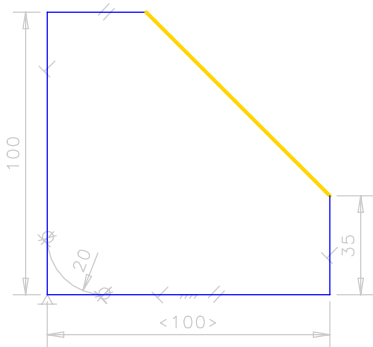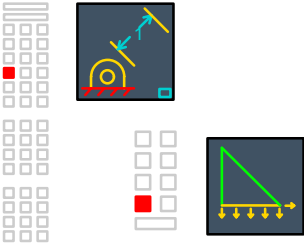

1

 (Done)

 Yes

 (to deactivate icon)

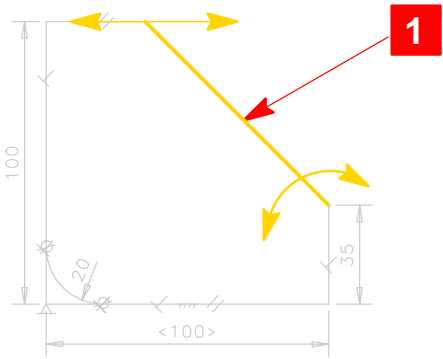Now, use the *Show Free* icon to find out what is not constrained.

Don't quit out of this command. You'll complete further steps on the next page.

Things to notice
Notice how the diagonal line is shown in yellow, indicating that it is not fully constrained.

1 pick the diagonal line

⬜🟦⬜ (Done)

Things to notice
The arrows show you the geometry that is still free to move.

⬜🟦⬜ (to deactivate icon)

Close the *Constrain* panel by double-clicking in the upper left corner.
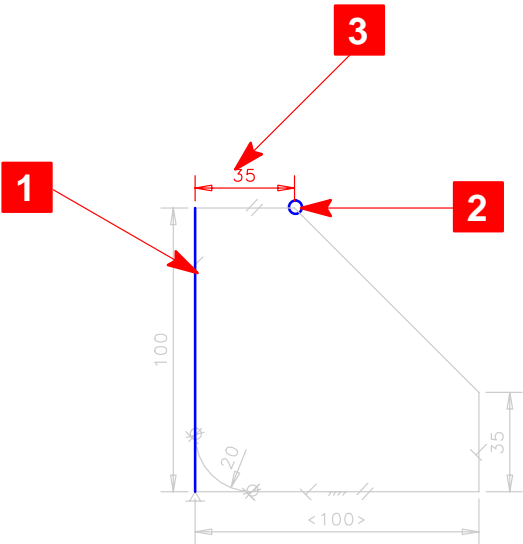
Recovery Point
▭ *File*
  Save

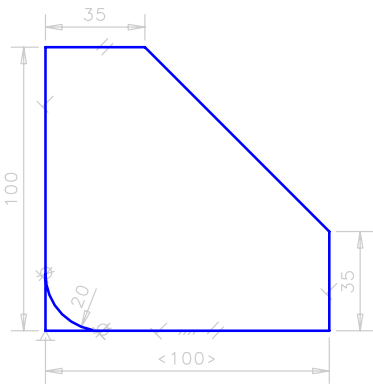Add a dimension that will fully constrain the geometry.

1 pick the line

2 pick the point
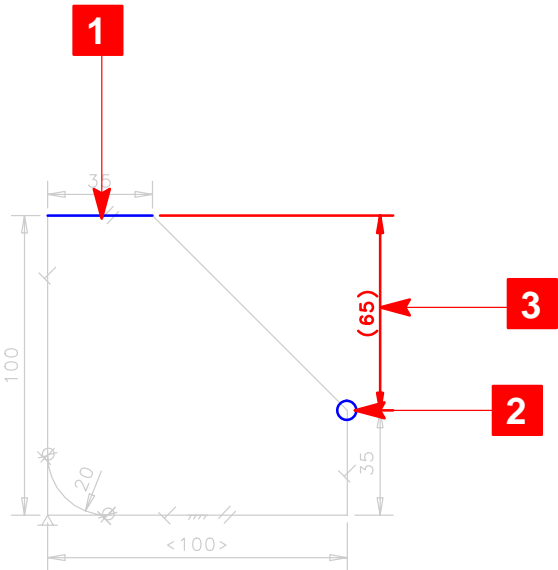
3 place the text

Things to notice

Notice how the line colors changed from yellow to blue as the dimension was added, since this dimension fully constrained the geometry.

Now, see what happens when you add a dimension to the fully constrained geometry.

1 pick the line

2 pick the point

3 place the text

Things to notice
Redundant dimensions are shown in parentheses and are called "reference" dimensions. They cannot be used to drive the geometry, but will change when other dimensions are modified.

Tutorial wrap-up

You have completed the Dimensioning tutorial.

Save the model file with this part. You'll use it in the next tutorial.